Chapter 9
Example Case 3

In this example, we will change the torsion-only shaft model from the previous example to a fully flexible model. This will demonstrate the required steps for replacing a section of the model which is internally connected to many other modeling elements.

9.1 Outline

NOTE: To work through this example, you should switch into the *nrel/examples/case_3* directory before starting ADAMS/View. As for the second example, assuming you have set up the environment variables and *aview.pth* file correctly, you can then start View and load the ADAMS/WT overlay by reading in the command file *wt_main.cmd*. (See Appendix I for more information on setting up WT.) Remember, the command for this is:

file command read fil=wt_main

At this point you should be ready to begin *case_3*.

1. Delete the default *hawt* model.
2. Load the *case_2* model from the command file.
3. Change the model name to *case_3*.
4. Delete the existing REQUESTs.
5. Delete the bearings and torsional spring from the low-speed shaft.
6. Change the names and ADAMS identifiers of the low-speed shaft PARTs.
7. Create a new, fully-flexible low-speed shaft.
8. Add the *hinge* MARKER to the *lss3* PART.
9. Transfer the motor-generator, hub spring and teeter stops to the new low-speed shaft.
10. Delete the old low-speed shaft PARTs.
11. Add new REQUESTs.
12. Do the analysis (using same executable and aero as for *case_2*).
13. Look at the results.

NOTE AGAIN: In order to avoid losing your work, we recommend that you save the ADAMS/View session to a binary file after each section in the example is completed. This can be done through the FILE SAVE menu, or from the command line in A/View by typing:

file binary write file=case_3 (or just *fil bin wri fil=case_3*)

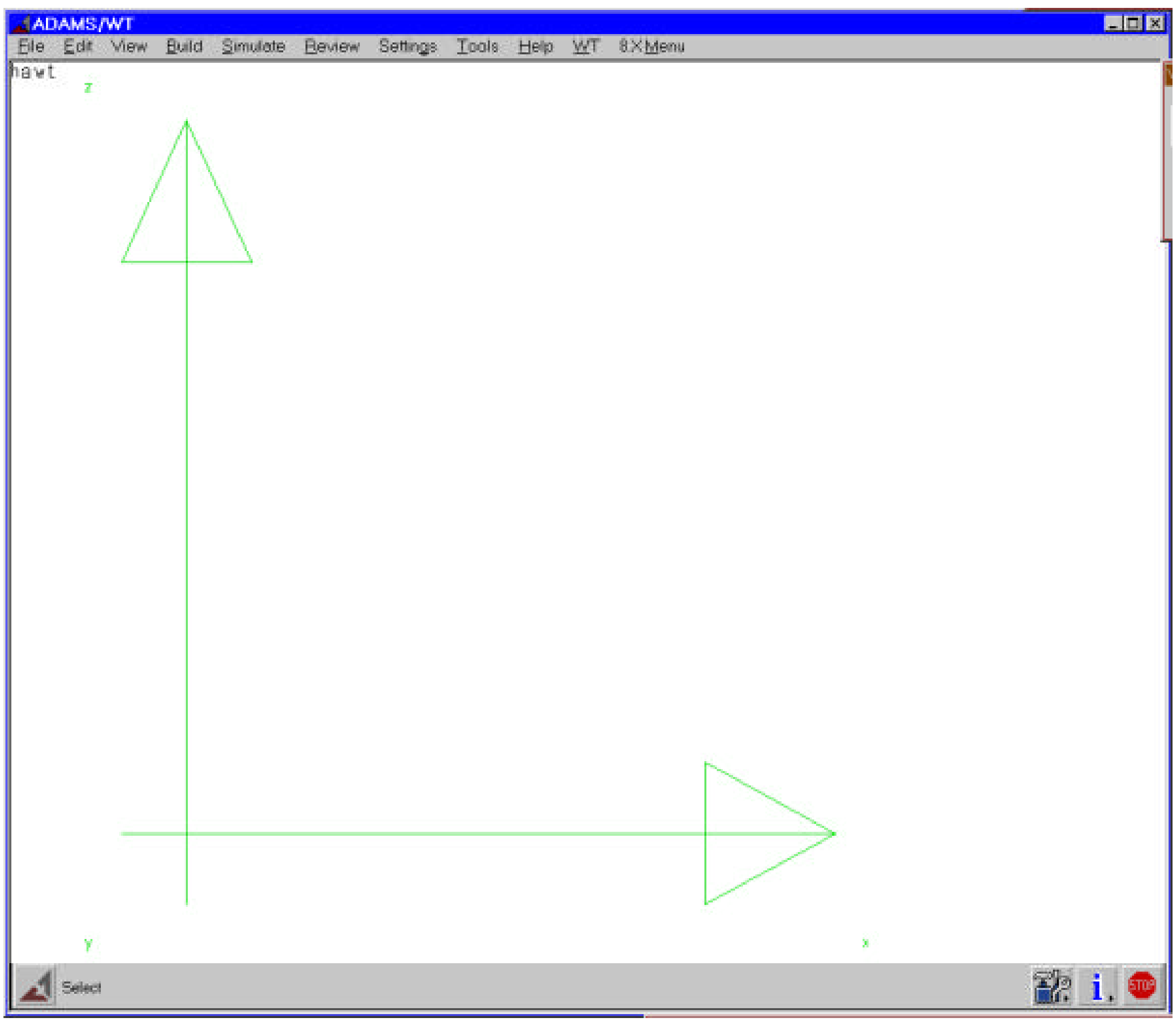
9.2 Startup

The input data for the *case_3* model is in the *examples/case_3* directory. In it you will also find an ADAMS/View command (*.cmd*) file for the *case_2* model. From the *examples/case_3* directory, bring up ADAMS/View by using the icon (NT) or by typing at the system command line:

mdi -c aview ru-s i ex & (UNIX)
or
mdi aview ru-s (NT)

Then start WT by reading in the *wt_main.cmd* command file (which loads the interface code) using the FILE IMPORT menu or from the View command line:

file command read file=wt_main



Next you should delete the *hawt* model, either through the main menu structure with BUILD MODEL DELETE or directly at the View command line with

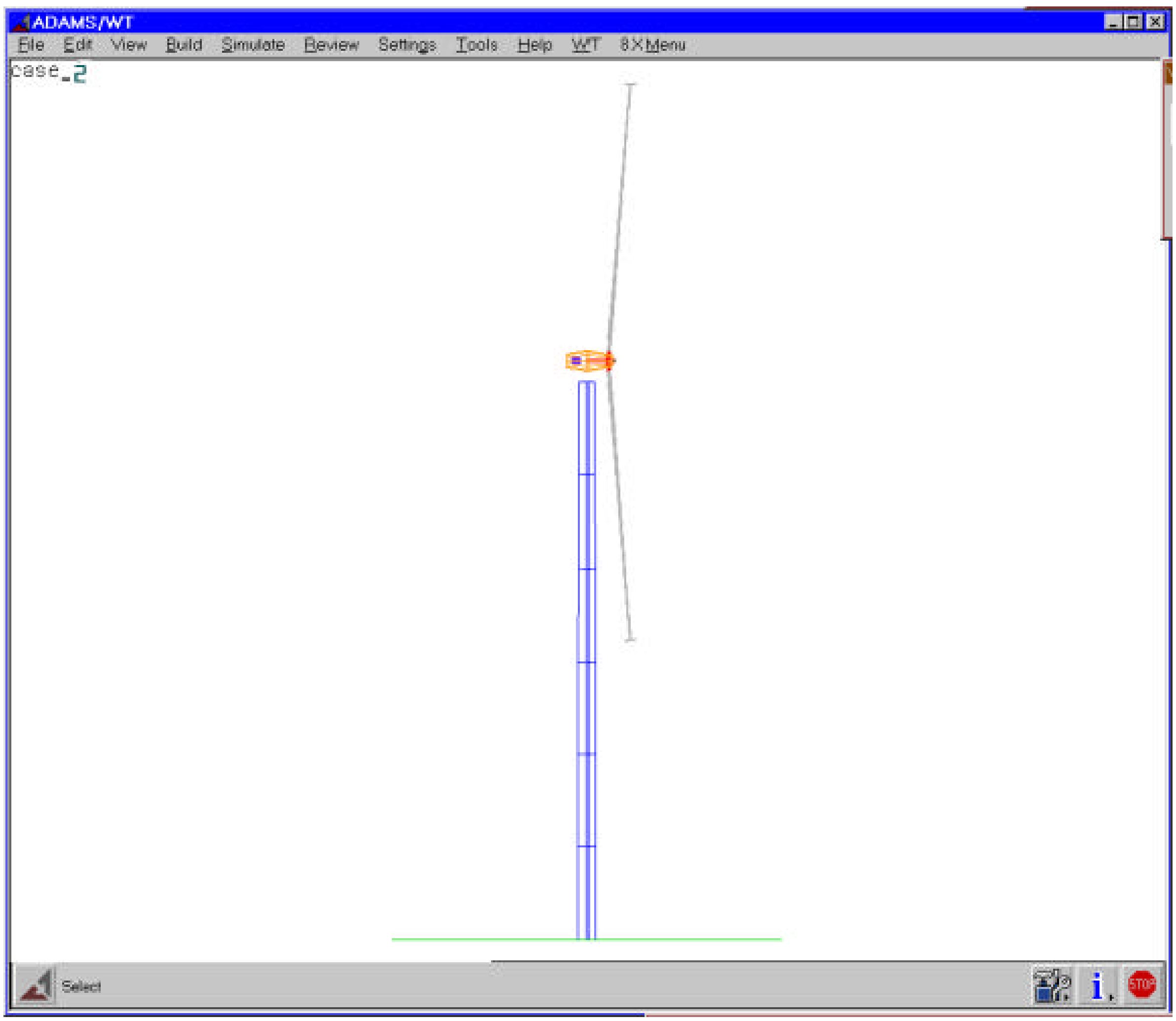
model delete model=hawt

This will leave you with a blank window, into which you can read the command file for the *case_2* example.

You should return to the View window and read the *case_2.cmd* file into View. This can be done through the menus using FILE IMPORT (recommended) or at the command line with:

file command read file=case_2

At this point you should have the *case_2* model displayed.



Now change the name of the model, either using the menu structure with BUILD MODEL RENAME, or at the command line with:

model modify model=case_2 new=case_3

At this point you are ready to start in on the necessary modifications. This would be a good place to save your work in a binary image.

9.3 Deleting Existing REQUESTs

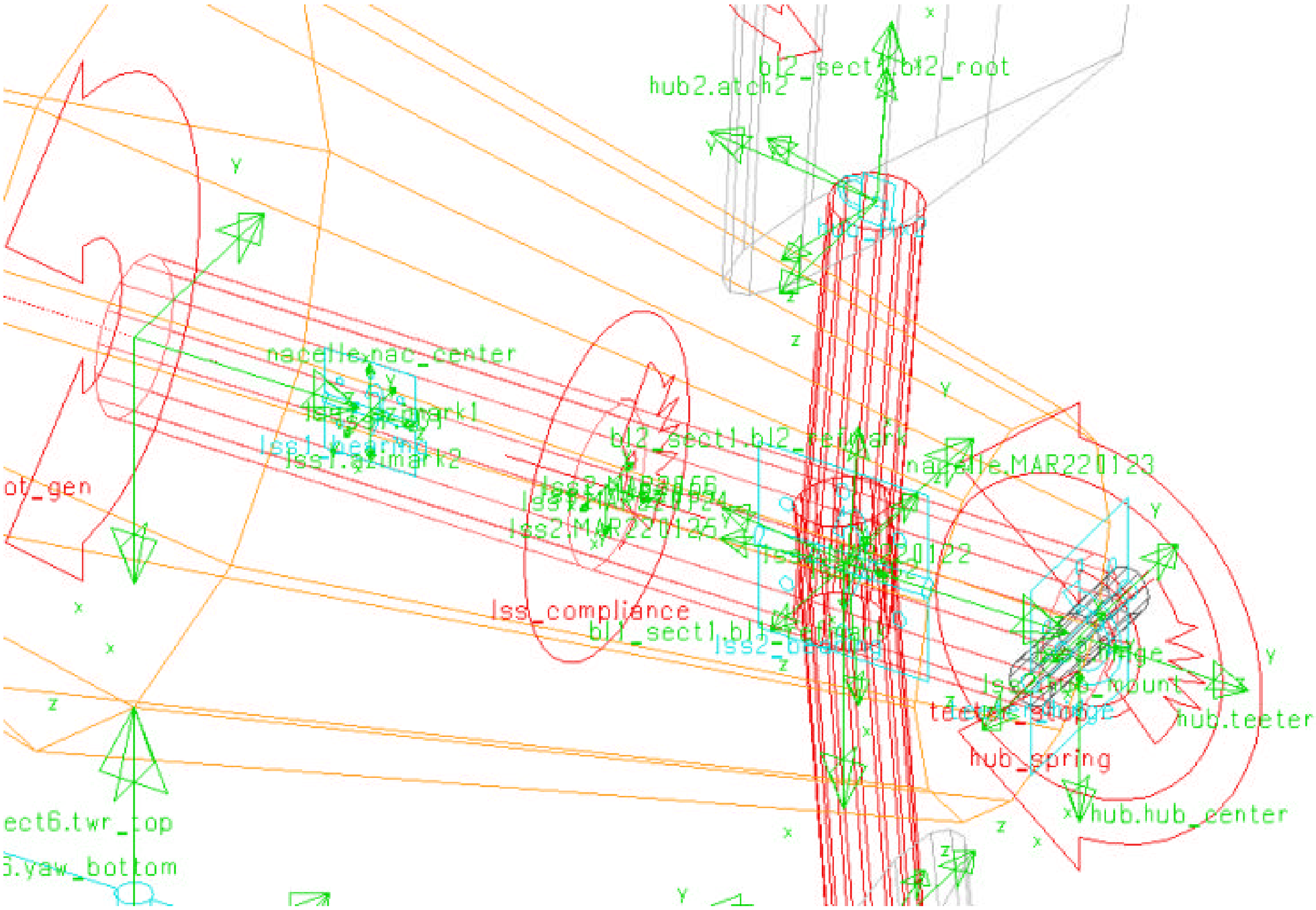
Before beginnning the model modifications, we must delete the exisiting REQUESTs because they are "connected" internally to various parts of the model. Unless the REQUESTs are deleted, A/View will not allow you to delete or modify those parts. Because ADAMS/View allows for wildcarding, deleting all the REQUESTs at one time is a very simple operation. It can be done through the Command Navigator by selecting OUTPUT_CONTROL DELETE REQUEST and entering an asterisk in the field, or directly from the command line in View by entering:

*output delete request request=**

9.4 Modifying Existing Low-Speed Shaft

Before the new low-speed shaft can be created, the old one must be modified so that the ADAMS/WT macro structure doesn't "recognize" it. There are two steps to this process. The first step is remove any "attachments" to the existing torsion-only low-speed shaft that will be superfluous for the new fully-flexible one. The second step is to rename and renumber the existing low-speed shaft PARTs so that the WT "standard" names and ID's for the shaft can be used on the new PARTs.

It will be very helpful to first turn on the icon visibility (from the Main Toolbox) and then adjust the view so that you have good access to the low-speed shaft area in the model. Using the PICK and ZOOM controls, you should get a view something like this:



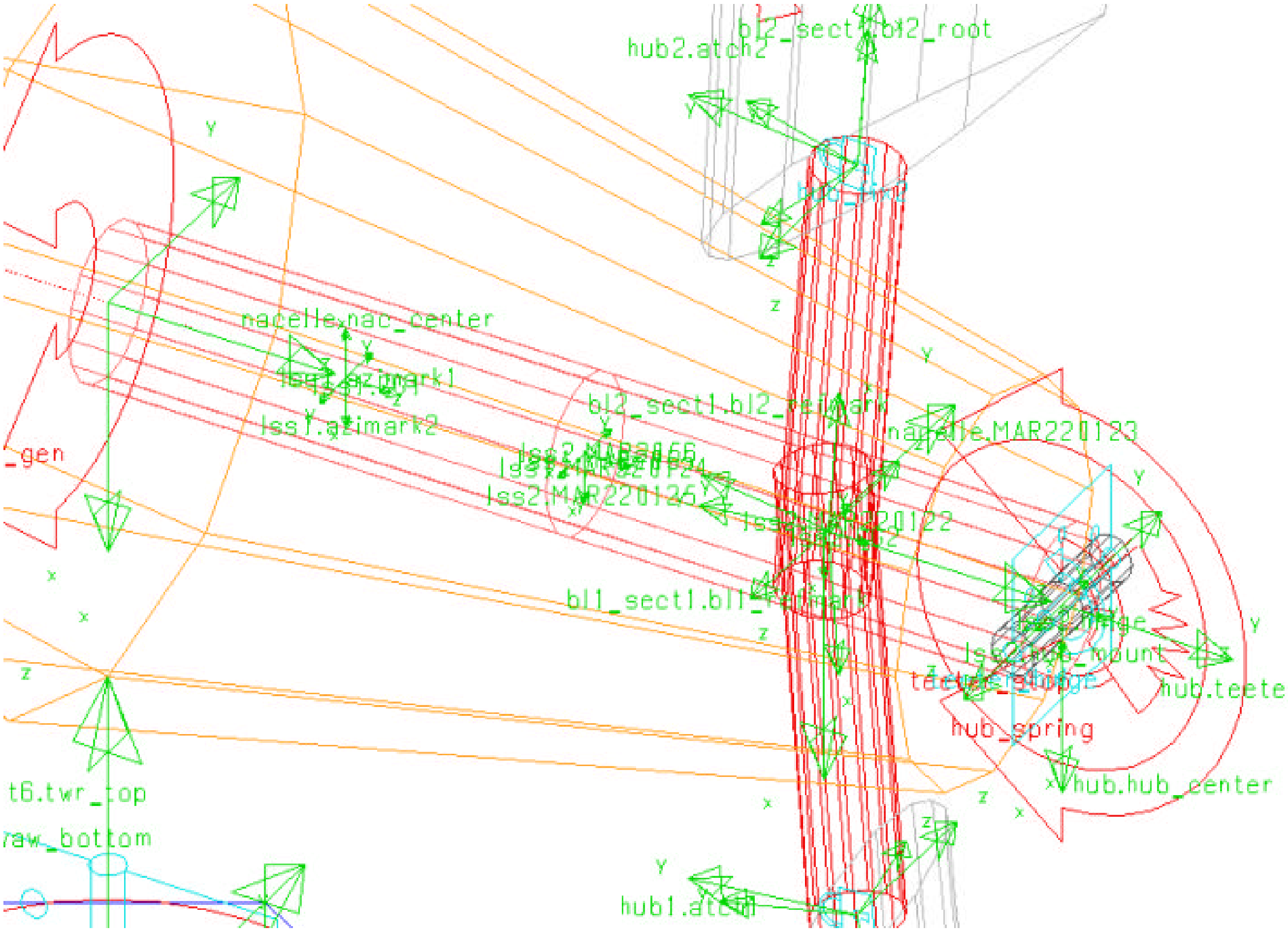
Then you can bring up the panel for deleting constraints from the 8.X Menu by selecting JOINT DELETE. You should then pick on the two shaft bearings and execute the panel. Alternately, you could enter at the View command line:

constraint delete constraint=lss1_bearing, lss2_bearing

Next you can bring up the panel for deleting forces from the 8.X Menu, by selecting FORCE ROTAT_SPRING DELETE. Pick on the torsional spring at the middle of the shaft and execute the panel. Alternately, you could enter at the View command line:

force delete force=lss_compliance

You model should now look something like this:



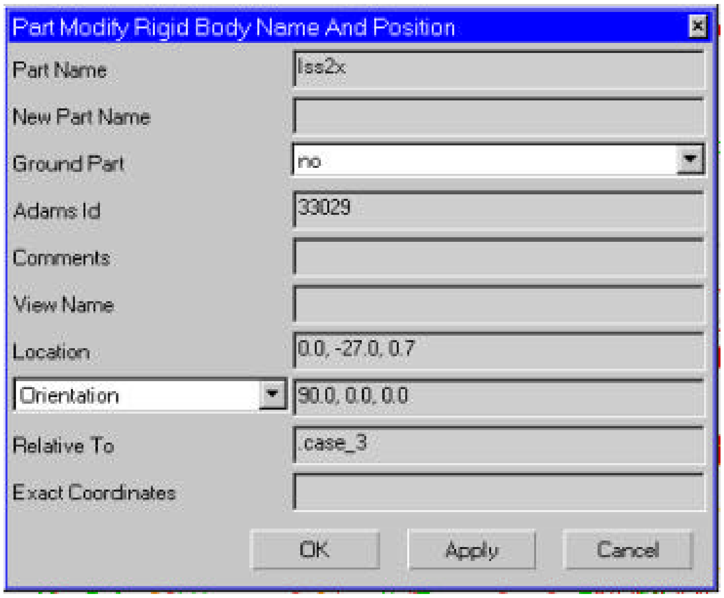
Next, you must modify the two low-speed shaft parts to give them different names and numbers which WT does not use nor expect to see. From the 8.X Menu, select the panel for PART MODIFY NAME_AND_POSITION. Pick *lss1*, or enter *lss1* in the part_name field, and hit Enter to load the panel with the correct information for the *lss1* PART. You should then change <u>only</u> the following data and Apply the panel.

new_part_name = lss1x
adams_id = 33019

You can then do the same operations for the *lss2* PART by going back to the part_name field and changing *lss1* to *lss2* and again hitting Enter to load the panel with the correct information for the *lss2* PART.. Then you should enter

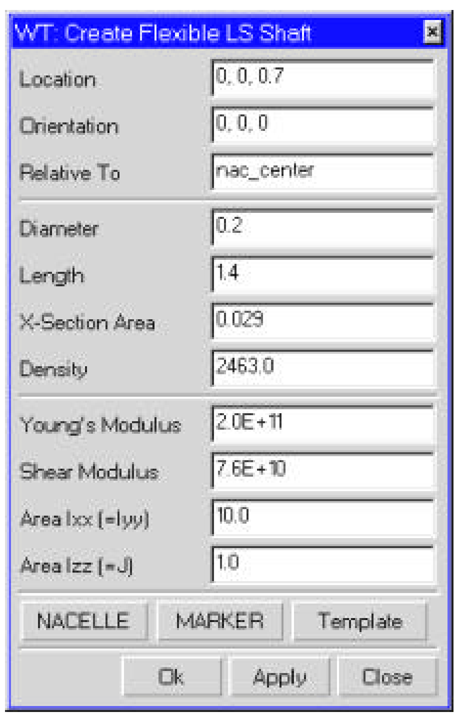
new_part_name = lss2x
adams_id = 33029

and execute the panel with OK. This would be good time to save your work.

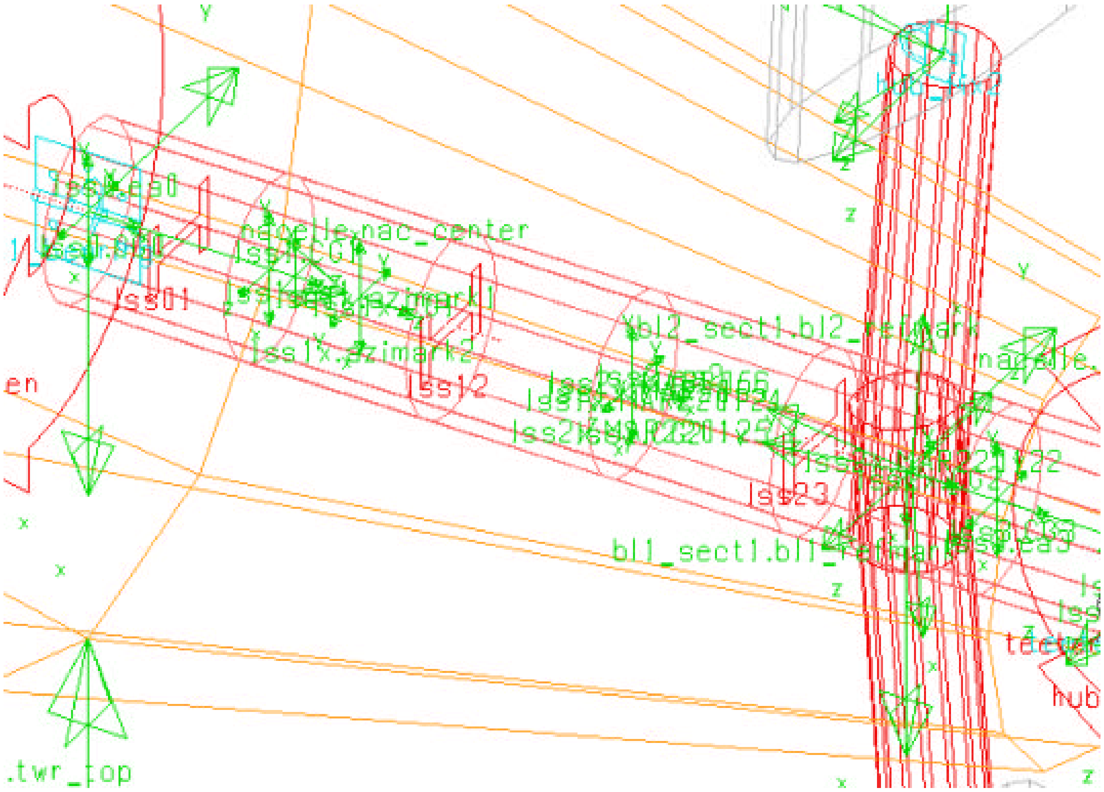


9.5 Creating the Flexible Low-Speed Shaft

Complete details of creating flexible low-speed shafts are found in section 5.4.4. The low-speed shaft panel is accessed through the WT menus by selecting POWER_TRAIN LOW_SPEED_SHAFT FLEXIBLE. When you do this, WT will bring up the visual template of the entire power train, and display the desired entry panel. Fill in the panel as shown.



This will build the new shaft directly over top of the old one.



9.6 Incorporating the New Low-Speed Shaft into the Model

Before the old low-speed shaft can be removed from the model, the other elements in the model which are attached to it have to be switched over to the new shaft. These elements include the *mot_gen*, *teeter_hinge*, *teeter_stop* and *hub_spring*. In addition, the two MARKERs which AeroDyn references for blade azimuth information have to be shifted to the new *lss0* PART. Finally, the VARIABLE named *slip* also depends on the old shaft parts and must be modified.

To change the motor-generator, you should bring up the modify panel for it through the 8.X Menu, selecting FORCE SFORCE MODIFY. Selct the *mot_gen* SFORCE by picking, browsing, guessing or just entering it directly. Then you should load the data for the force element by hitting Return. You should see

i_marker_name = .case_3.lss1x.CG1

Change this to

i_marker_name = CG0

and then execute the panel. There will be no apparent change at all in the view, but the *mot_gen* force no longer acts on the old shaft.

Before you can switch the *teeter_hinge*, *hub_spring* and *teeter_stop* to the new shaft, you must create a MARKER to which they can connect. Through the 8.X Menu, select MARKER CREATE. Enter the following data and execute the panel by selecting OK.

marker_name = .case_3.lss3.hinge
...
location = 0,0,0
orientation = 0,0,0
relative_to = teeter

Amidst all the clutter at the end of the shaft, you may not notice the new MARKER! Regardless, you can now modify the remaining connections.

Bring up the panel for modifying the *teeter_hinge* from the 8.X Menu with JOINT REVOLUTE MODIFY. Pick on the *teeter_hinge* icon in the view, and it should auto-load the joint data. You should see:

j_marker_name = case_3.lss2x.hinge

Change this to

j_marker_name = case_3.lss3.hinge

and execute the panel. Again, you will see no obvious change in the View.

We need to do the same basic operation for the two force elements. First, from the 8.X Menu, select FORCE ROTAT_SPRING MODIFY to bring up the panel for modifying the *hub_ spring* element. Pick on the *hub_spring* icon (looks like /\/\/\/\) and it should auto-load the spring data.. Then change the j_marker_name field just like you did for the *teeter_hinge*, so it reads

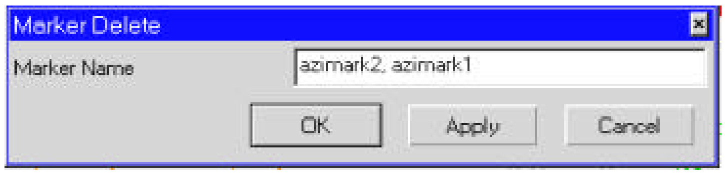
j_marker_name = case_3.lss3.hinge

and execute the panel.

Finally, select FORCE SFORCE MODIFY to bring up the next needed panel. Again. pick on the *teeter_stop* SFORCE in the view and it should auto-load the force data.. For this force, you must change <u>both</u> the j_marker_name field <u>and</u> the function text. In both places, change all references to *lss2x* to references to *lss3*, just as you did previously. It may be easiest to do this using the A/View Function Builder, which you can bring up by right-clicking in the function field.

The last change we need to make is to the *slip* definition. From the main menu, select BUILD SYSTEM_ELEMENT STATE_VARIABLE MODIFY. This will bring up the Database Navigator. Find the *slip* VARIABLE and double-click on it to get the Modify panel. Then right-click on the function field to get into the View Function Builder. You should see a long expression including a reference to .case_3.lss1x.CG1. You must replace that reference with .case_3.lss0.CG0. Hit the OK button to load the corrected expression into the function field, and then execute the Modify panel to incorporate the changes. You can split the lines if necessary to avoid warnings about "... line longer than 80 characters... "

The last operation required before removing the old shaft is to delete the existing AeroDyn azimuth MARKERs (so you can re-use the ADAMS identifiers) and create new ones on the *lss0* PART. From the 8.X Menu, select MARKER DELETE and in the marker name field, enter *azimark1, azimark2* as shown.



To create the new MARKERs, from the 8.X Menu, select MARKER CREATE. For the first one, enter these data:

marker_name = .case_3.lss0.azimark1
adams_id = 3051
location = 0,0,0
orientation = 0,0,0
relative_to = nac_center

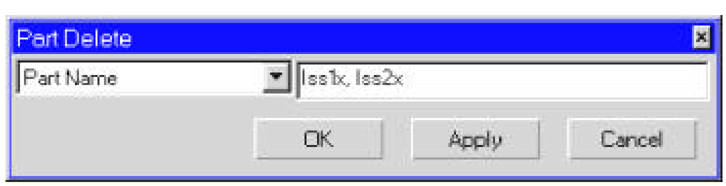
and Apply the panel. For the second MARKER enter these data:

marker_name = .case_3.lss0.azimark2
adams_id = 3052
location = 0,0,0
orientation = 0,0,180
relative_to = nac_center

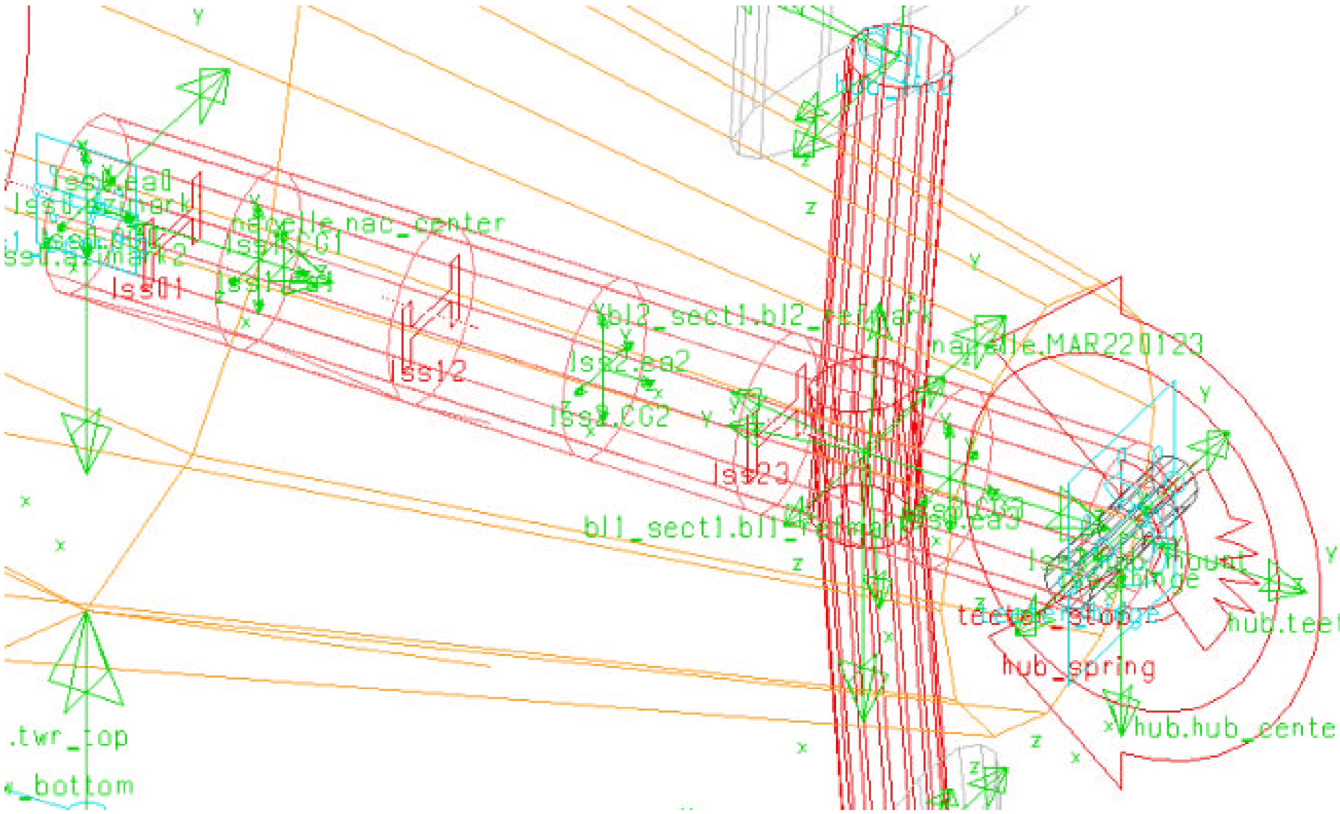
and execute the panel with OK. This completes the required adjustments to the new shaft and would be a good time to save your work.

9.7 Removing Old Low-Speed Shaft

This can be very easily done from the 8.X Menu. Select PART DELETE and in the part name field enter *lss1x, lss2x*



Your drive train should now look like this:



9.8 Output Requests

The automatic output requests can be regenerated much as they were for example case #1 (see section 7.12). Return to the main WT menu, select REQUESTS, and check each of the different kinds of output you can solicit from ADAMS.

For the MOTOR-GENERATOR request, you must define the two MARKERs between which the *mot_gen* force acts, You should enter

M-G attachment marker on stator = *.case_3.stator.CG*
M-G attachment marker on LSS = *.case_3.lss0.CG0*

For the LOW-SPEED SHAFT, you must specify which beam element in the shaft you want to monitor. WT will create both force-type and displacement-type requests based on the I and J markers for that element. For this example, choose the *lss12* BEAM by clicking on it in the View window or by just entering it in the field.

Which compliant element in the LSS = *lss12*

For the BLADE_ROOT request, you must specify which blade you want to monitor.

Which blade? = 1

When you have completed all the automatic REQUESTs, you may add additional REQUESTs to the model using any and all of the methods allowed by ADAMS, i.e. standard type requests, functionally defined requests or REQSUB user-subroutine-generated requests. This is a good spot to again save your work.

9.9 Running the Simulation

As long as you don't specifically change the settings, the *case_*3 model <u>should</u> inherit the same configuration of results *(.res),* output *(.out),* request *(.req)* and graphics *(.gra)* file output, but you may want to check them from the WT-Specific Extras menu. Select Results File Controls, and set

Create Results File =	off
Formatted =	off

Then select General Output Controls and set

Reqsave =	on
Grsave =	on
everything else =	off

It should also <u>not</u> be necessary to change any of the integrator and solver parameters for this analysis.

At this point, your model is complete and you should write it out in dataset (*.adm*) format for simulation and in View binary (*.bin*) and command file (*.cmd*) formats for safekeeping. These actions may be accomplished from the FILE SAVE and EXPORT menus, or from the A/View command line. In the command line window, you can type:

file adams write file=case_3
file command write file=case_3 entity=case_3
file binary write file=case_3

Now you are ready to try your first simulation of the *case_3* turbine. You should copy, link or move the *wt20.exe* file you created in the first example problem into the *examples/case_3* directory for use with this model. If you did not do the first example, refer now to section 7.13 for instructions on how to build the user-executable version of ADAMS/Solver.

You must again create an ADAMS/Solver command file *(.acf)* to control the simulation. Using your editor, create a text file named *case_3.acf* with the following contents:

```
case_3
case_3
integrator/gstiff,err=.01
sim/dyn,end=0.25,step=50
integrator/err=1e-3
sim/dyn,end=0.5,step=50
integrator/err=1e-4
sim/dyn,end=1,step=100
sim/dyn,end=1.5,step=100
sim/dyn,end=2.0,step=100
sim/dyn,end=2.5,step=100
sim/dyn,end=3,step=100
sim/dyn,end=3.5,step=95
sim/dyn,end=4,step=95
sim/dyn,end=5,step=180
sim/dyn,end=6,step=180
sim/dyn,end=7,step=180
sim/dyn,end=8,step=180
sim/dyn,end=9,step=180
sim/dyn,end=10,step=180
stop
```

To run the code you can again use the menu interface step by step, or enter the single long command at the system prompt:

mdi -c ru-u i wt20.exe case_3.acf exit

or for NT

mdi ru-u wt20.exe case_3.acf

At this point, ADAMS/Solver should start up and the simulation progress should be displayed on screen. The looser error tolerance as start up is to avoid some difficulty with simulation startup, but as long as the tolerance is reduced again, the overall response should be good. The program log is also written to the file *case_3.msg*. When the run is complete, you should be returned to the system prompt and the simulation results should be in the files *case_3.gra* and *case_3.req*. The *.msg* file should contain something very similar to the previous runs and is not reproduced for this case. Note that this model with the flexible low-speed shaft is generating many more output steps (about 2° steps) , is much more highly coupled, and will take much longer to run than the previous cases.

9.10 Visualizing the Results

At this point, you are ready to read the results of the simulation back into ADAMS/View to look at the responses. Switch back to the A/View window and either use the FILE IMPORT menu or enter at the View command line:

file analysis read file=case_3 model=case_3

It will take View a few moments to read in the data from the graphics (*case_3.gra*) and request (*case_3.req*) files. You can then animate the results and see how the rotor responded. There are quite a few ways to animate response in View. The simplest way in the WT interface is to bring up the Main Toolbox and just hit the ANIMATE button.

9.11 Plotting Output

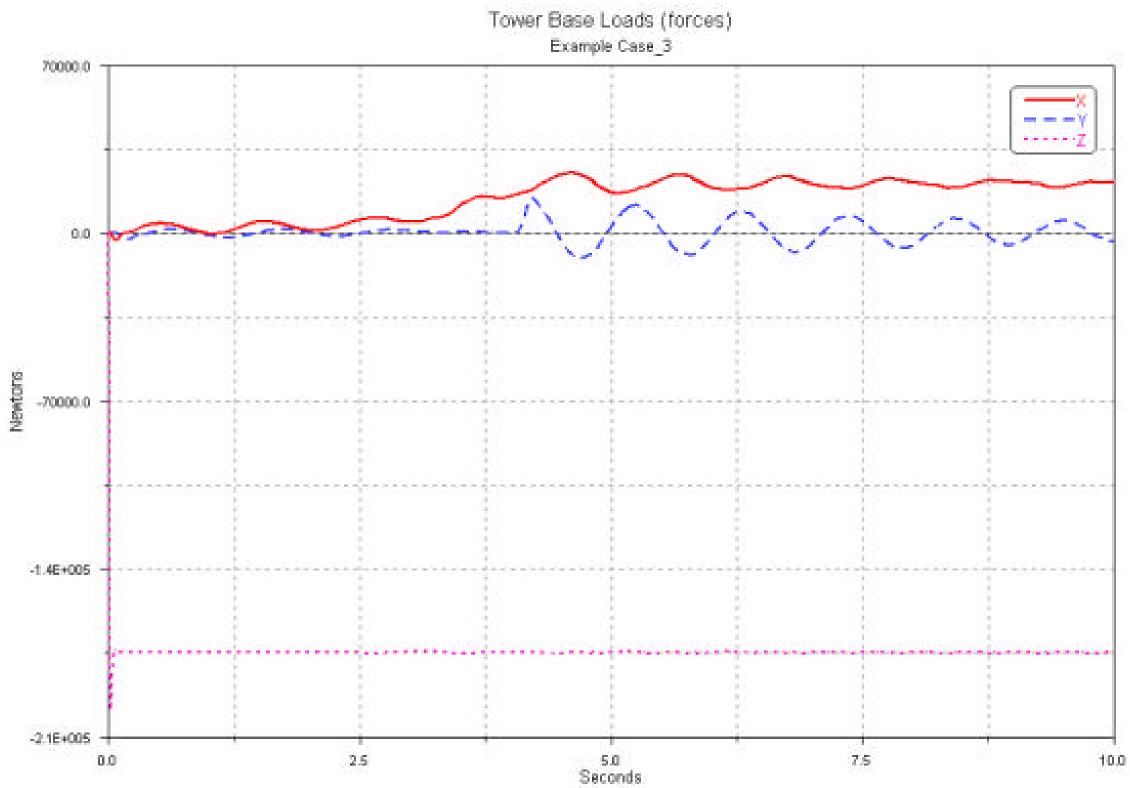
ADAMS/View 9.1 has a completely new plotting interface, including a large number of plotting features which can be accessed in many ways. Quick plots of request data can be made by easily made using the Plot Builder. The data can also be "surfed" this way.

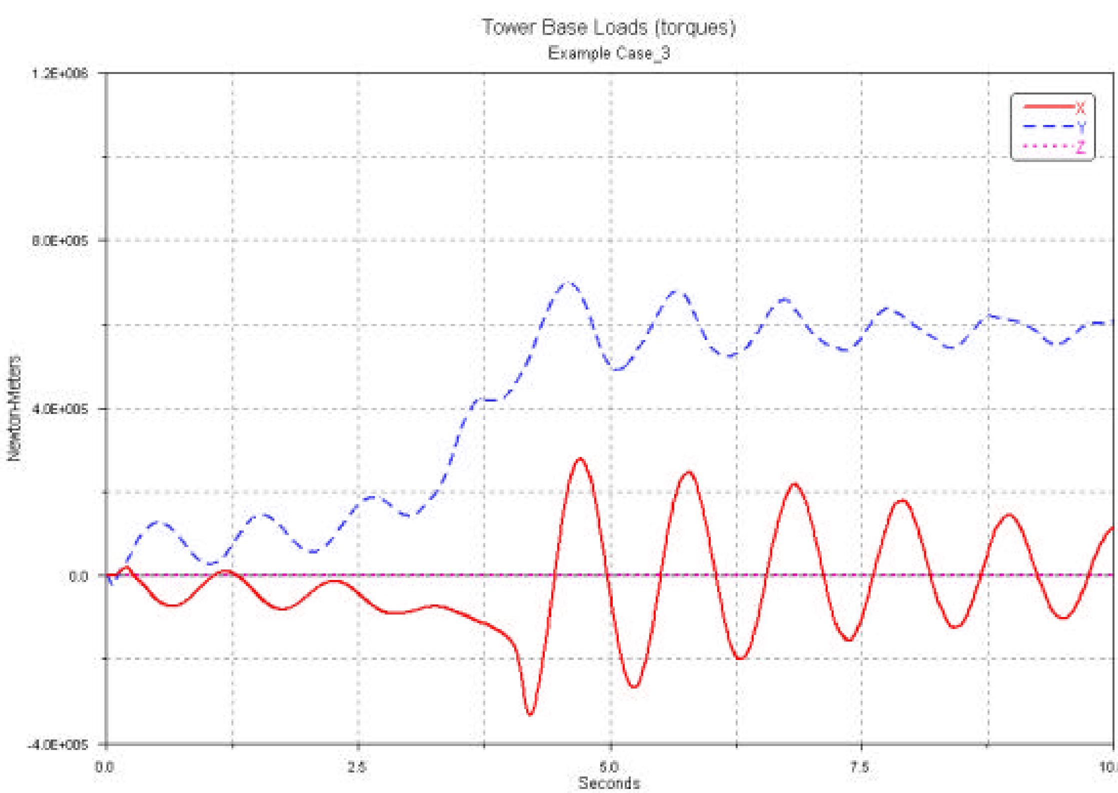
For repetitive plotting of specific requests from multiple simulations, it is often best to create a View command file (*.cmd*) containing the necessary commands to create and customize all the plots for a particular run. This command file contains the same commands you could execute via the plot builder or type in at the View command line to create the plots, but is easily modifiable using a text editor for customization and changes. An example of such a command file is found in the file *plotemup.cmd* in the *examples/case_3* directory. The contents are listed in detail in section 7.17, with the only differences being a change between Case 1 and Case 3 in the subtitle and the addition of the plot of the torsional response of the shaft..

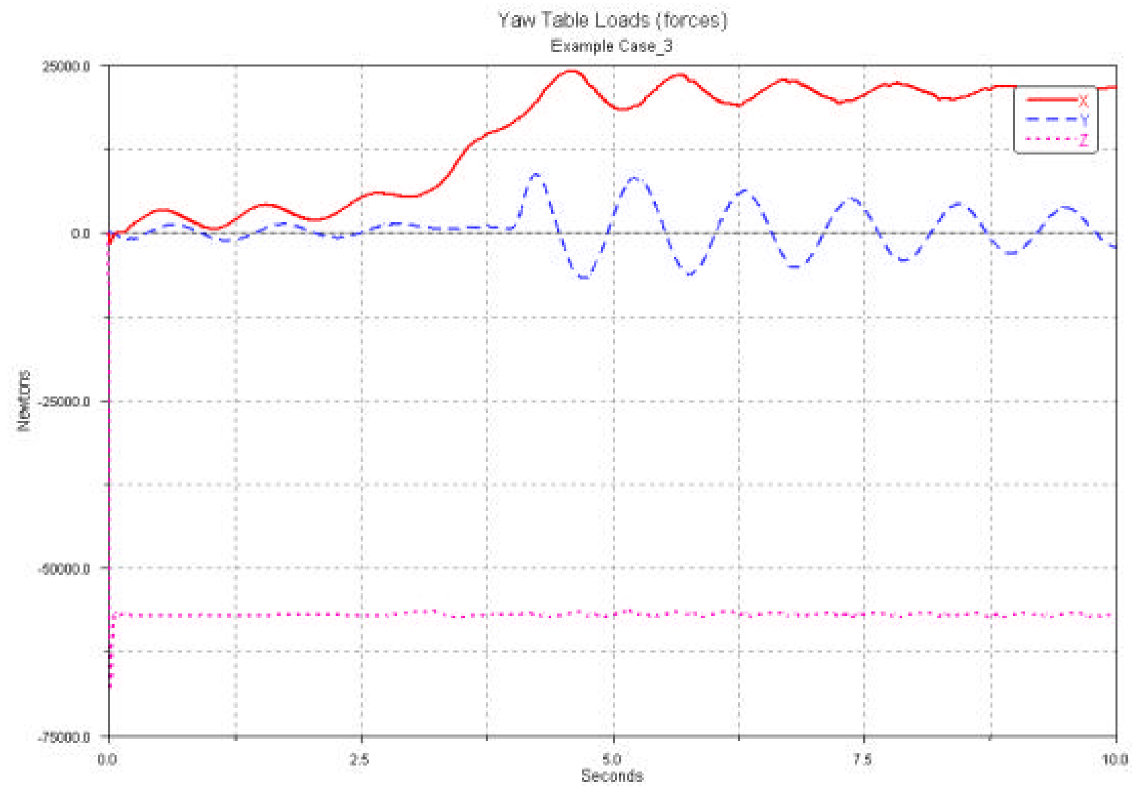
You can read in and run this command file through the FILE IMPORT menus or by entering at the View command line:

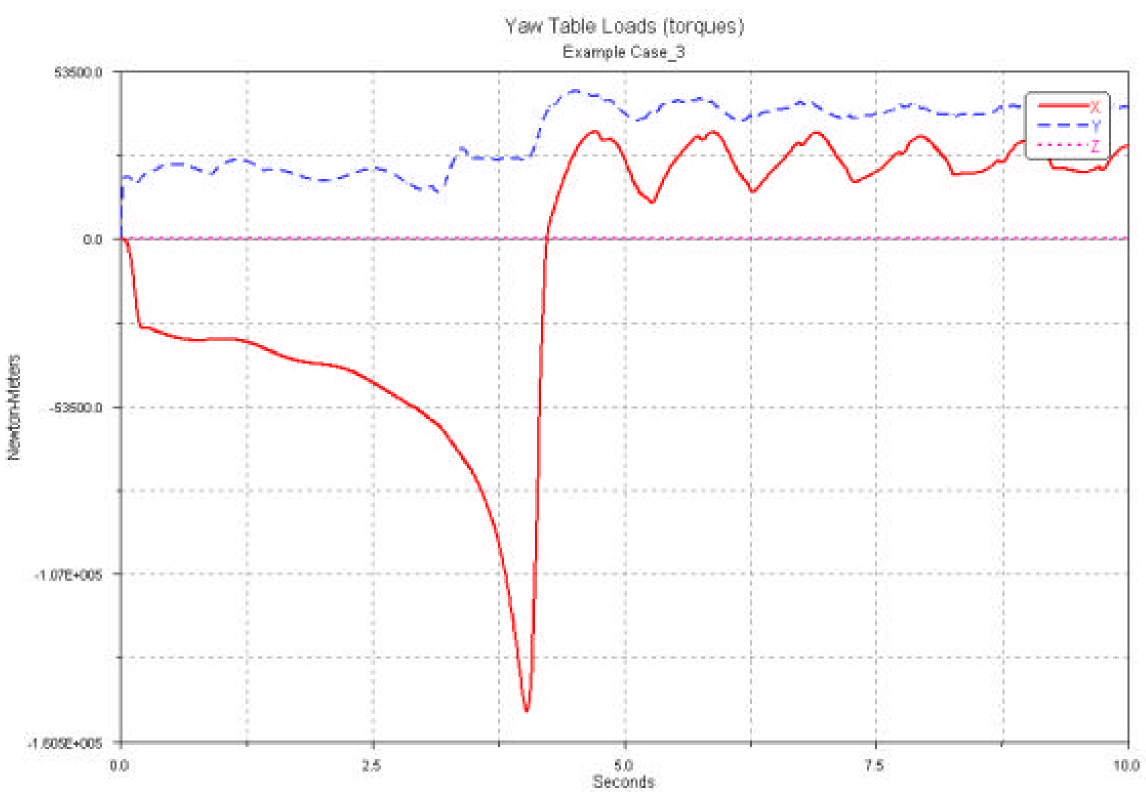
file command read file=plotemup

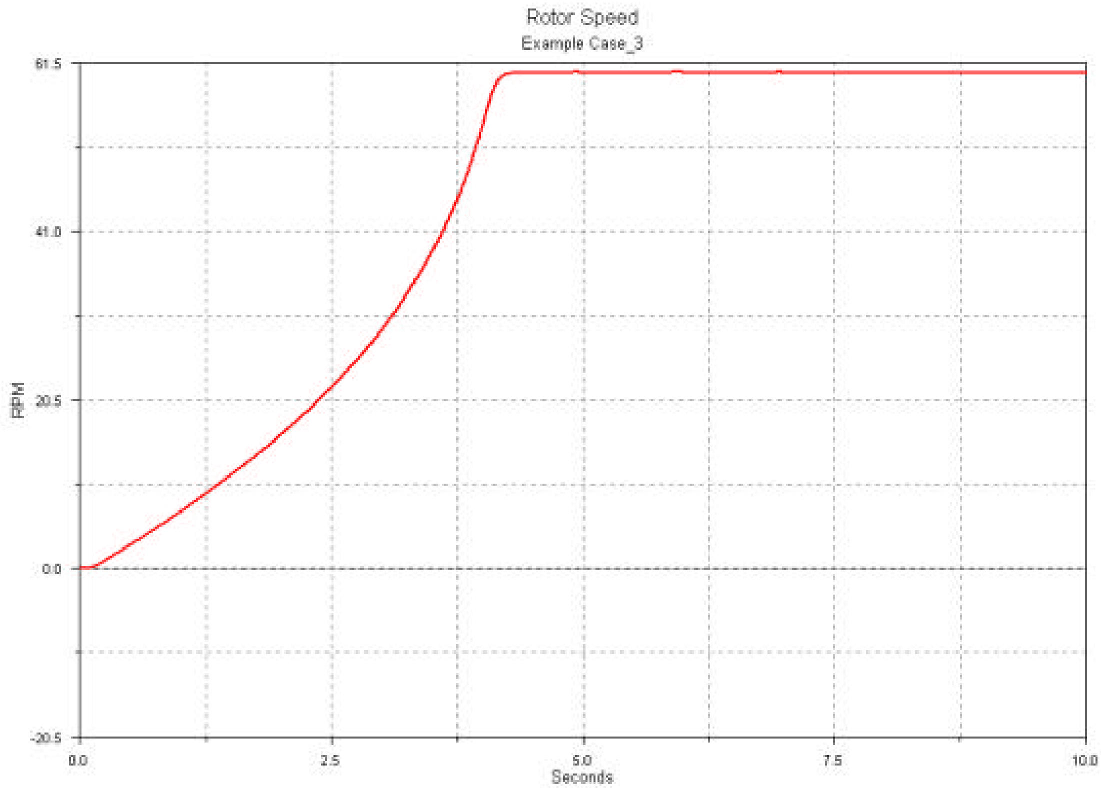
The example plots below can be used to confirm that your model and WT executable are working correctly.

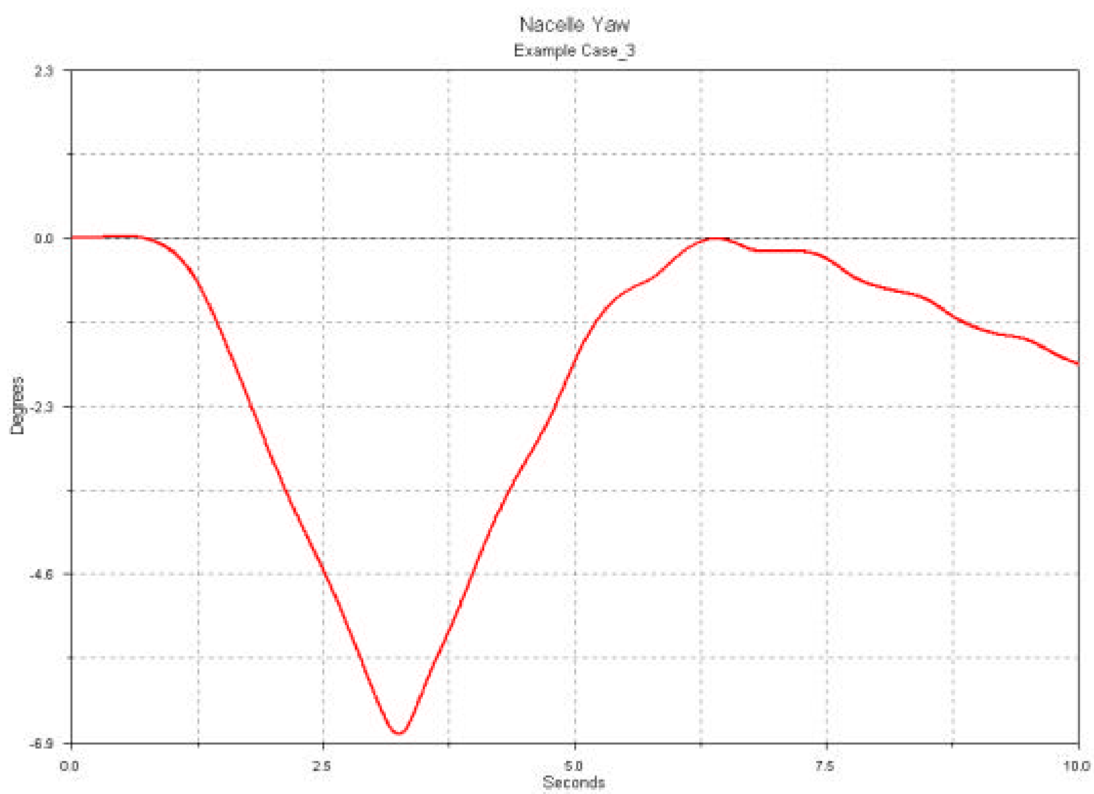

Tower Base Loads (forces)
Example Case_3
X
Y
Z
70000.0
0.0
-70000.0
-1.4E+005
-2.1E+005
Newtons
0.0
2.5
5.0
7.5
10.0
Seconds

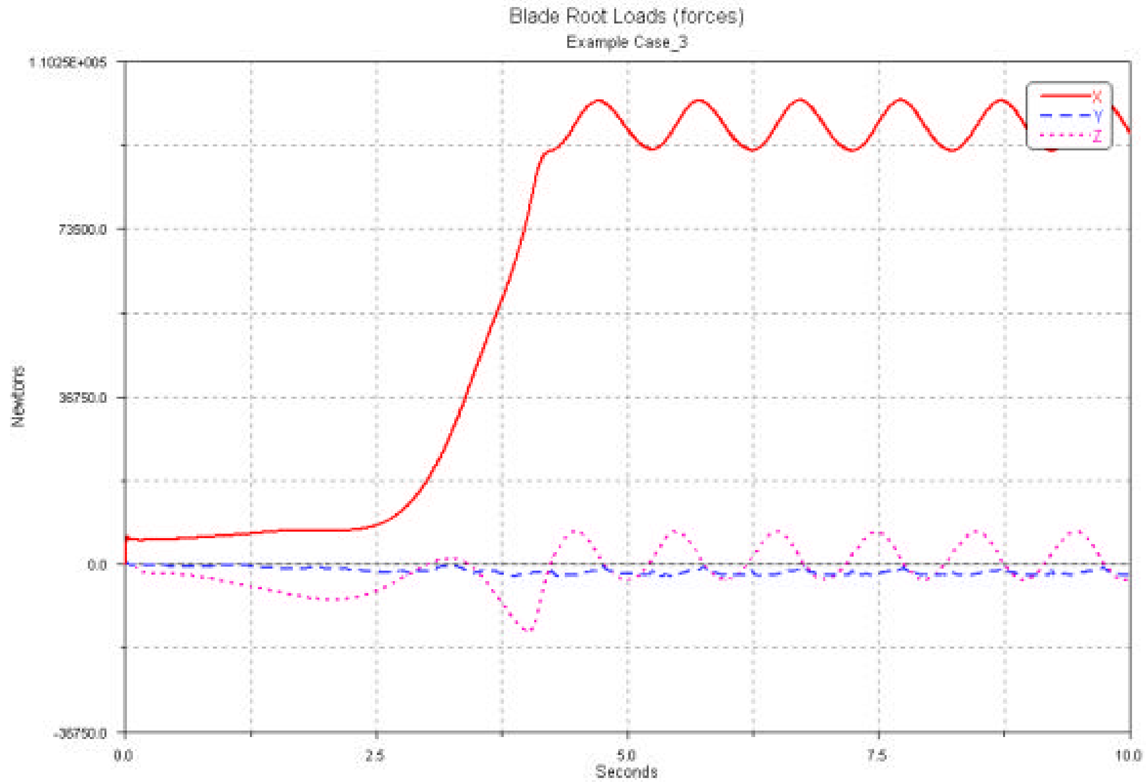

Tower Base Loads (torques)
Example Case_3
X
Y
Z
1.2E+006
8.0E+005
4.0E+005
0.0
-4.0E+005
Newton-Meters
0.0
2.5
5.0
7.5
10.0
Seconds

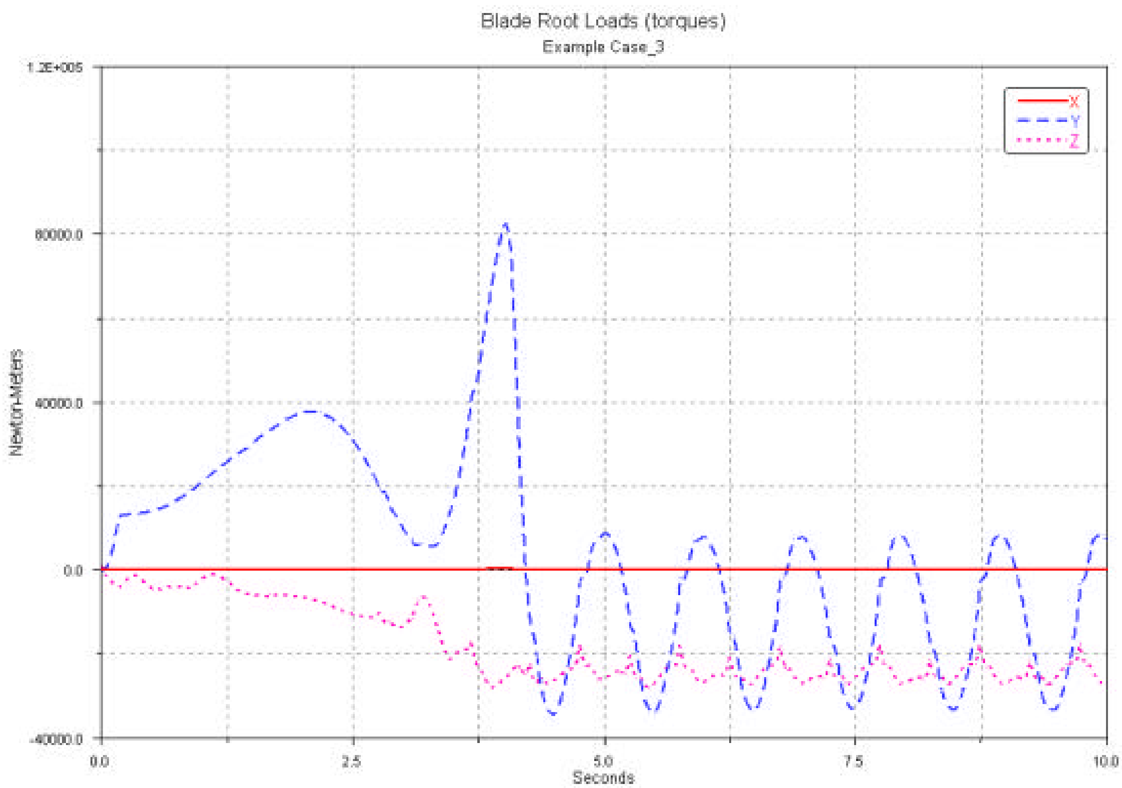

Yaw Table Loads (forces)
Example Case_3
25000.0
0.0
-25000.0
-50000.0
-75000.0
Newtons
0.0
2.5
5.0
7.5
10.0
Seconds
X
Y
Z

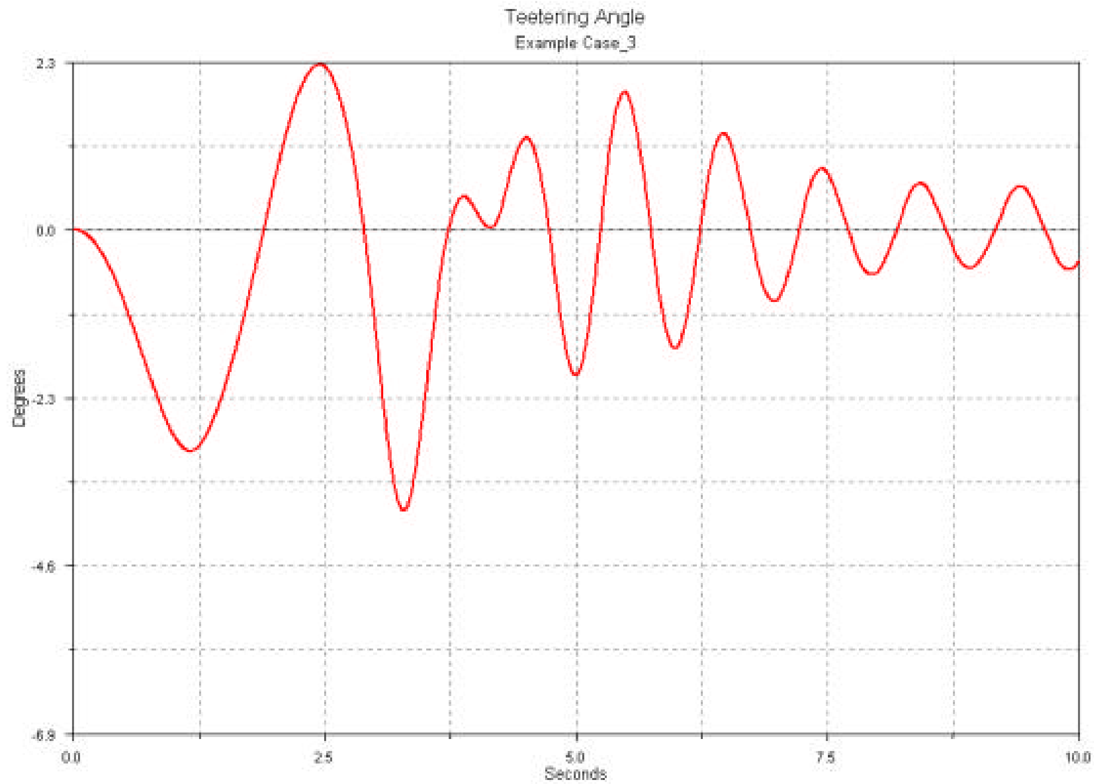

Yaw Table Loads (torques)
Example Case_3
53500.0
0.0
-53500.0
-1.07E+005
-1.605E+005
Newton-Meters
0.0
2.5
5.0
7.5
10.0
Seconds
X
Y
Z

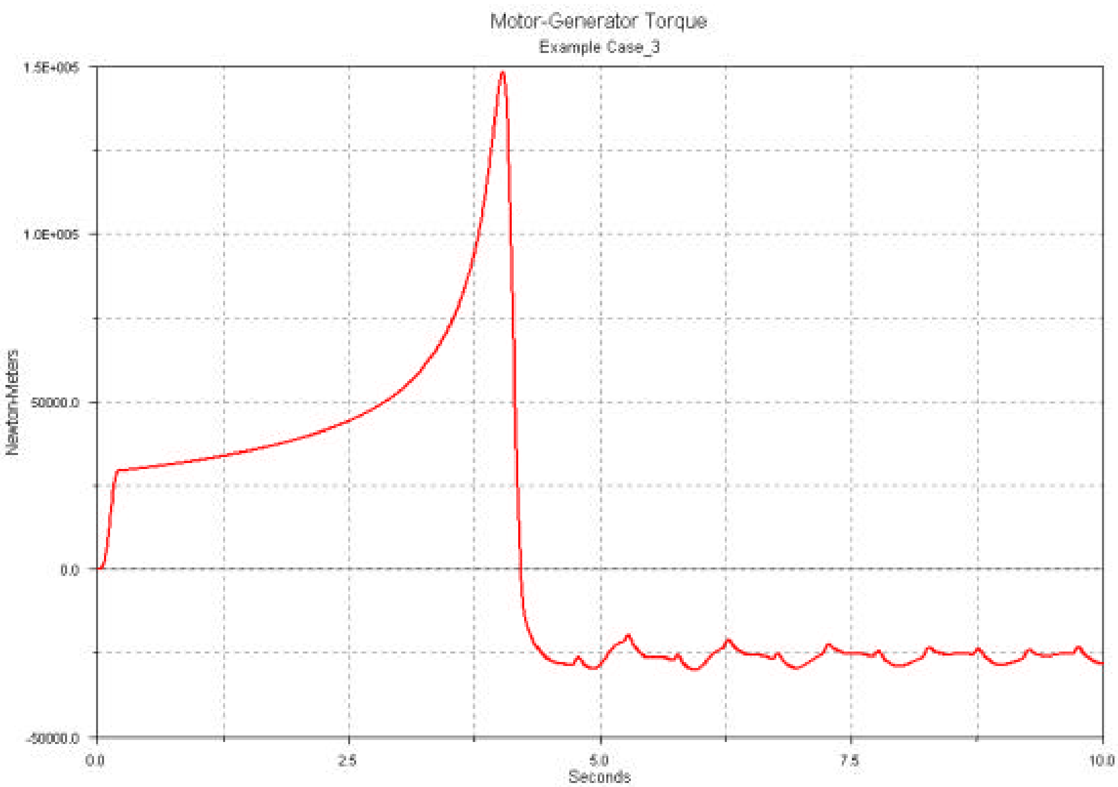

Rotor Speed
Example Case_3
61.5
41.0
20.5
0.0
-20.5
RPM
0.0
2.5
5.0
7.5
10.0
Seconds

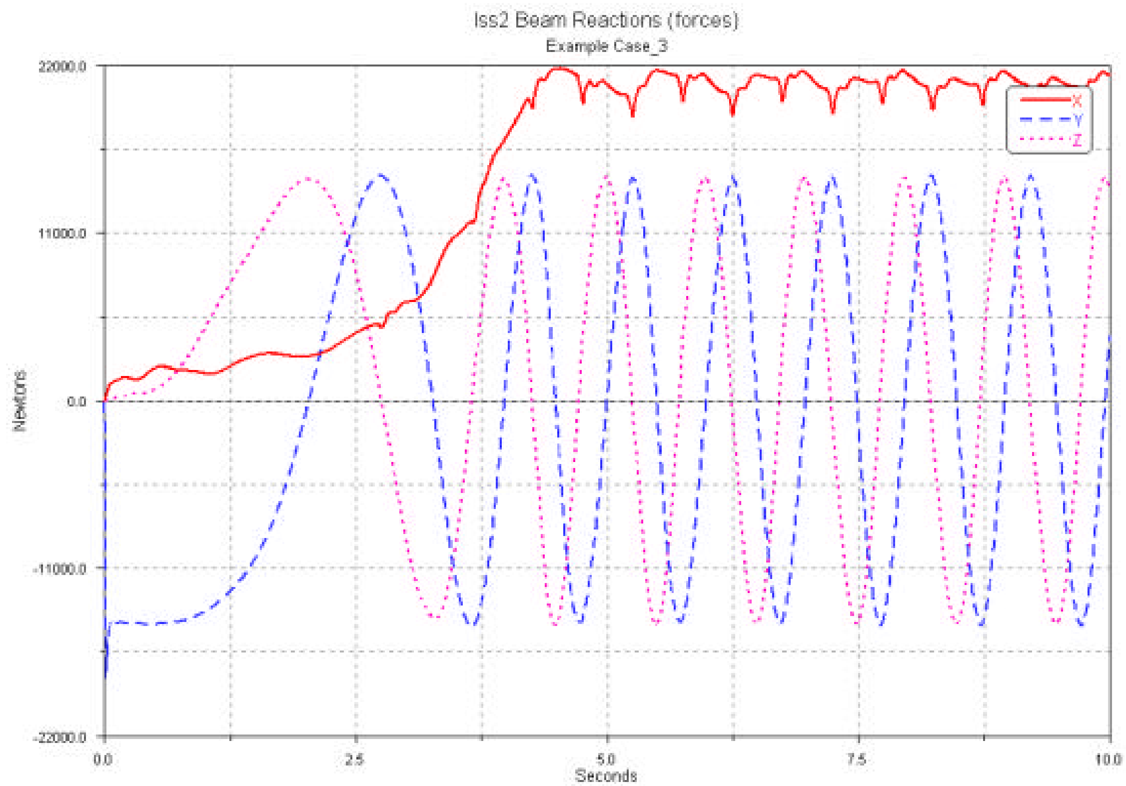

Nacelle Yaw
Example Case_3
2.3
0.0
-2.3
-4.6
-6.9
Degrees
0.0
2.5
5.0
7.5
10.0
Seconds

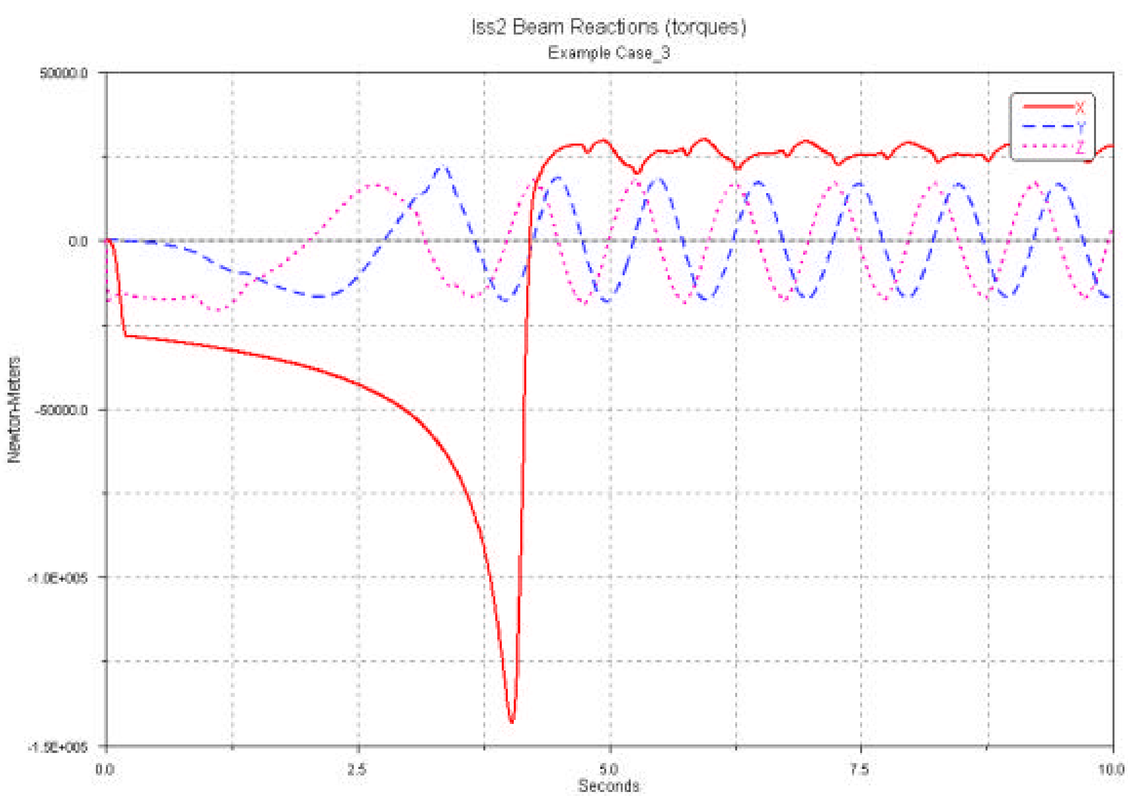

Blade Root Loads (forces)
Example Case_3
1.1025E+005
73500.0
36750.0
0.0
-36750.0
Newtons
0.0
2.5
5.0
7.5
10.0
Seconds
X
Y
Z


Blade Root Loads (torques)
Example Case_3
1.2E+005
80000.0
40000.0
0.0
-40000.0
Newton-Meters
0.0
2.5
5.0
7.5
10.0
Seconds
X
Y
Z


Teetering Angle
Example Case_3
Degrees
Seconds
2.3
0.0
-2.3
-4.6
-6.9
0.0
2.5
5.0
7.5
10.0


Motor-Generator Torque
Example Case_3
Newton-Meters
Seconds
1.5E+005
1.0E+005
50000.0
0.0
-50000.0
0.0
2.5
5.0
7.5
10.0


Iss2 Beam Reactions (forces)
Example Case_3
22000.0
11000.0
0.0
-11000.0
-22000.0
Newtons
0.0
2.5
5.0
7.5
10.0
Seconds
X
Y
Z


Iss2 Beam Reactions (torques)
Example Case_3
50000.0
0.0
-50000.0
-1.0E+005
-1.5E+005
Newton-Meters
0.0
2.5
5.0
7.5
10.0
Seconds
X
Y
Z